MIE346 – Eagle

This document is a quick reference for using Eagle to eventually create printed circuit board (PCB) layouts and test renders. It is not comprehensive, but the design tools are advanced enough that the end result is a mostly ready-to-manufacture PCB. If you are interested in creating your own boards for any personal or school projects, feel free to contact the instructor for additional resources.

Installation

You will need to install the free version of Eagle:

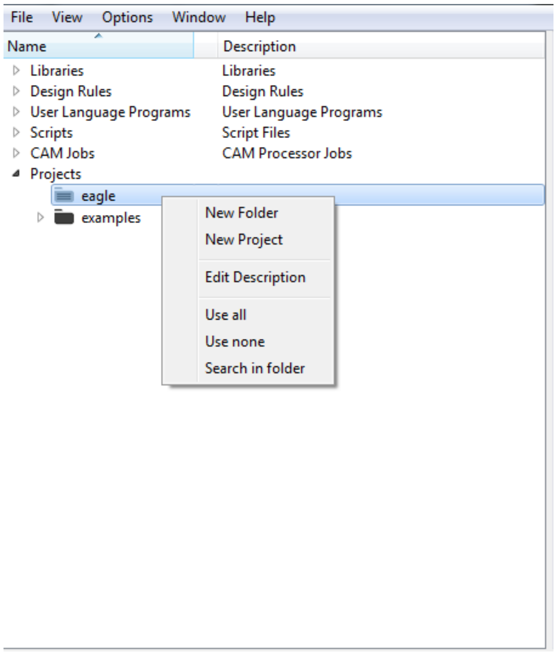
http://www.cadsoftusa.com/download-eagle/
http://web.cadsoft.de/ftp/eagle/program/7.2/eagle-win-7.2.0.exe (Windows Installer)

NOTE: *The current version of Eagle (top link) is 8.1.0. This tutorial was written originally for 7.2.0 (which is directly linked). If you choose to use the most recent version, some button and menu locations may change. The overall set of steps will remain very similar, though. We recommend simply using 7.2.0 to avoid any possible issues if you are unfamiliar with the software.*

No special options need to be selected during installation; follow the on-screen instructions. Select the free license when asked. This software can directly create schematics and PCB layouts.

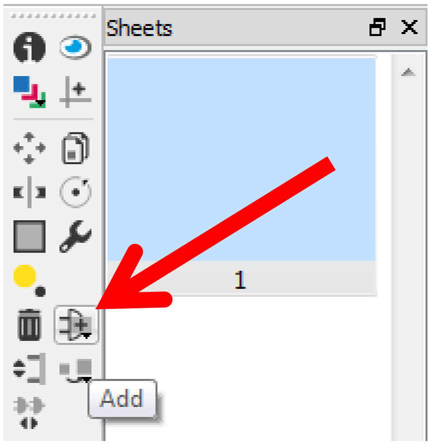
Creating a Schematic

The steps to creating a schematic diagram in Eagle are very similar to those used in PSPICE, although this software does not offer simulation capability. To begin, open Eagle from your programs. It will open with the Control Panel window:

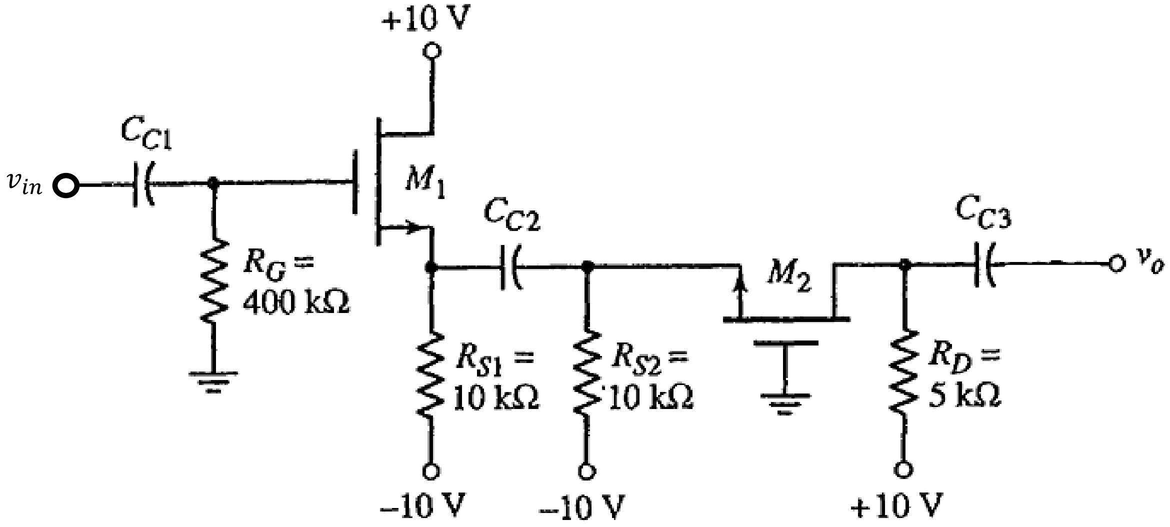


Name	Description
Libraries	Libraries
Design Rules	Design Rules
User Language Programs	User Language Programs
Scripts	Script Files
CAM Jobs	CAM Processor Jobs
Projects	
eagle	
examples	

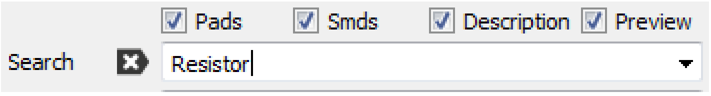
Under the Projects → 'eagle' section, create a 'New Project' with any name that you choose – we will use 'TestAmplifier' for this example. Right-click on the new project and select 'New Schematic' to create a new schematic. This will open a new, blank schematic window – you may want to save this schematic now with the same name as the project – 'TestAmplifier.sch' in our case. We will begin by adding parts to our schematic. Select the 'Add Parts' icon on the tool panel:



This will bring up a searchable list of parts. For this tutorial, we are going to use one of the past exam circuits as an example:



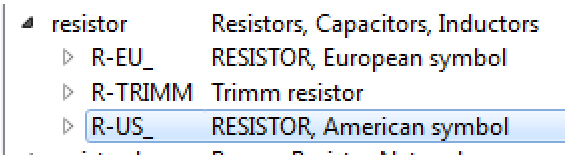
We can begin by adding the resistors. They can be found by searching ‘resistor’:



☑ Pads ☑ Smds ☑ Description ☑ Preview

Search Resistor

A large number of results will be returned. Although the resistor symbol is standard, the physical device may be packaged in a number of different ways. We select the American Standard packages:



resistor	Resistors, Capacitors, Inductors
R-EU_	RESISTOR, European symbol
R-TRIMM	Trimm resistor
R-US_	RESISTOR, American symbol

Within this category, there are a number of choices, each corresponding to a different mounting style (surface mount versus through-hole) or package size (5mm, 10mm, 15mm lead spacing, etc.). Assuming that we are using resistors similar to those in the labs, we will use the 10mm lead spacing (1/4 W power) through-hole resistor. This is one area where further work would be needed for a real board: the chosen outline would need to match the actual, physical part you will eventually use. If we assume the same resistor type as the lab components, we select the R-US/0207 option for all resistors. You can make the same choice for your assignment, or you can select a different through-hole type to match a physical component you chose.

Parts in your MIE346 lab component kit

The following is a list of parts that are available in your parts kit and their equivalent part names is Eagle. You may use these parts in your designs if you so choose.

Part	**Library Location** (search for:)
All Resistors	R-US_0207/10
6.8uF Capacitor	C-US050-035X075
0.082uF Capacitor	C-US075-032X103
LM324AN	LM324N
2.2uF Capacitor	CPOL-USE2-5
4.7uF Capacitor	CPOL-USE2-5
10uF Capacitor	CPOL-USE2-5
100uF Capacitor	CPOL-USE2.5-6
IRF510	IRF510
1N4001	1N4004
1N4733A	1N4728
LM78L05	78L05Z
Binding Post	Binding Post (part used by staff in reference designs, footprint available for download on course website)

If you are unsure about a part, you can always add it for now, view the actual PCB footprint it generates, and modify it at a later time.

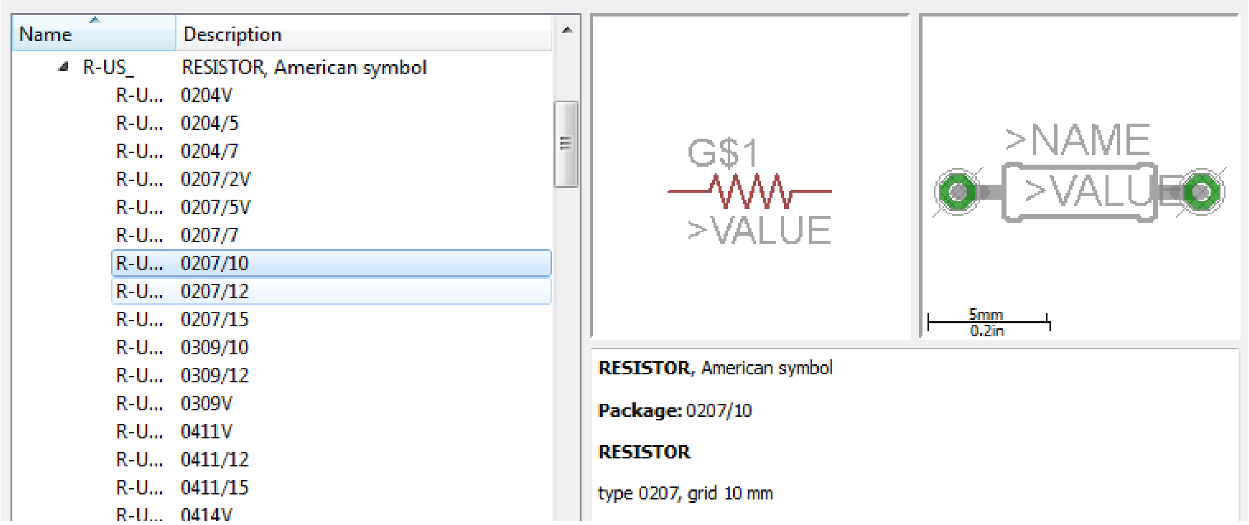
How to import parts into Eagle

Although Eagle comes with a large number of parts searchable through its libraries, occasionally the footprint/symbol for a part may need to be loaded from an external source. In such cases, library files will be distributed, such as for the 'Binding Post' (the banana plug socket on the reference designs). The file 'binding-post_10.lbr' can be downloaded from the course website, and it contains the schematic symbol and footprint for this part.

1. To use the library, save the 'binding-post_10.lbr' file in somewhere you can easily locate.
2. In the Schematic window, select the 'Library' menu, followed by the 'Use…' option.
3. In the selection window that opens, browse to the file you saved and open it.
4. Now, when you press the 'Add Part' button, the part will be available in the list of parts (under the name 'binding-post_10' → BINDING-POST).

You can repeat the above steps for any other part libraries you may find online, but note that the vast majority of designs should be able to be completed with the built-in libraries only.

Returning to the Example



Name	Description
R-US_	RESISTOR, American symbol
R-U...	0204V
R-U...	0204/5
R-U...	0204/7
R-U...	0207/2V
R-U...	0207/5V
R-U...	0207/7
R-U...	0207/10
R-U...	0207/12
R-U...	0207/15
R-U...	0309/10
R-U...	0309/12
R-U...	0309V
R-U...	0411V
R-U...	0411/12
R-U...	0411/15
R-U...	0414V

RESISTOR, American symbol

Package: 0207/10

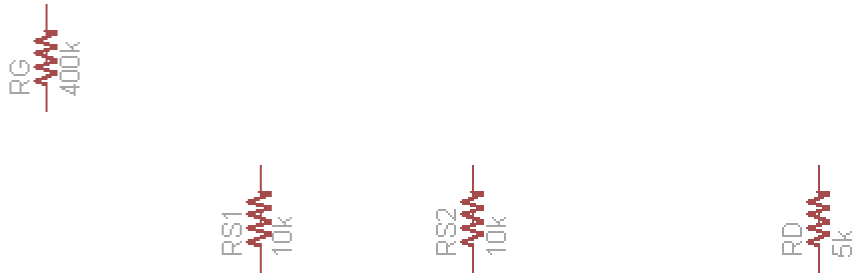
RESISTOR

type 0207, grid 10 mm

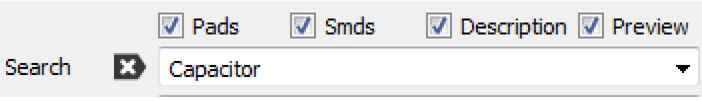
We now place resistors in all locations needed on our diagram. You can continue to left-click to add more resistors when in the 'Add Part' mode. When done, press the Escape key twice (once returns you to the parts list if you want to add more parts). A few notes on manipulating schematics:

- If you want to move a part, either select Edit→Move, or the move icon on the toolbar (). Then, left-click the part you want to move and left-click again to place it at its new position.
- The rotation tool (or Edit→Rotate) and the mirror tool (or Edit→Mirror) work in the same way.
- You can right-click on a component and select 'Name' to change its schematic name (R1, R2, etc.) and 'Value' to change the component value. Note that the software uses the same suffixes for value as PSPICE: $p, n, u, m, \ , k, M$. You can also remove the component by selecting 'Delete.'

Manipulate the resistors until they resemble the schematic, including names and values:

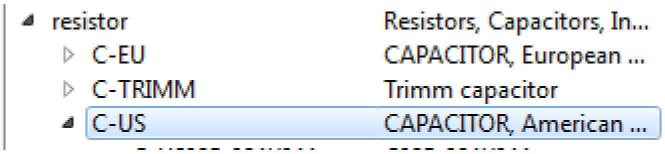


Next, add new parts and search for ‘Capacitor’ and select the American symbols:



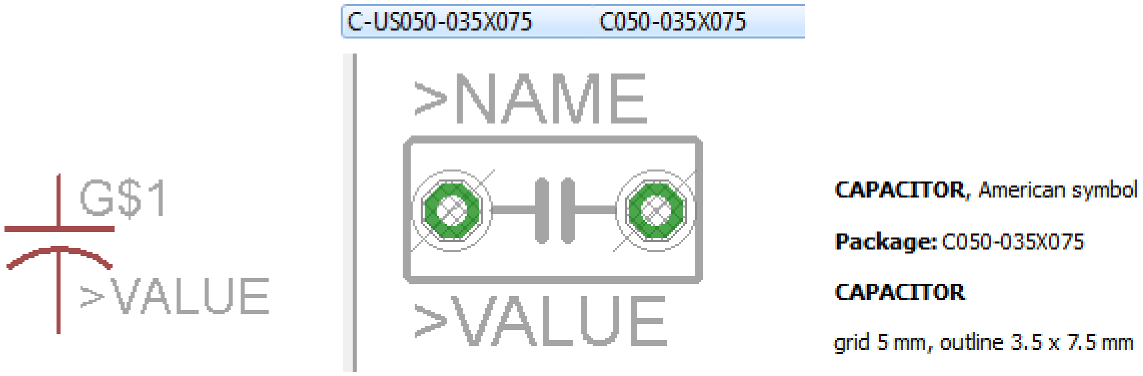
☑ Pads ☑ Smds ☑ Description ☑ Preview

Search Capacitor



resistor	Resistors, Capacitors, In...
C-EU	CAPACITOR, European ...
C-TRIMM	Trimm capacitor
C-US	CAPACITOR, American ...

Once again, in a real design you would select a capacitor footprint to match the physical size of the real-world device you select. We will assume that all capacitors are large and have a value of $10\mu F$. This matches the lab capacitors you used – the physical footprint for these capacitors is C-US050-035X075:



C-US050-035X075 C050-035X075

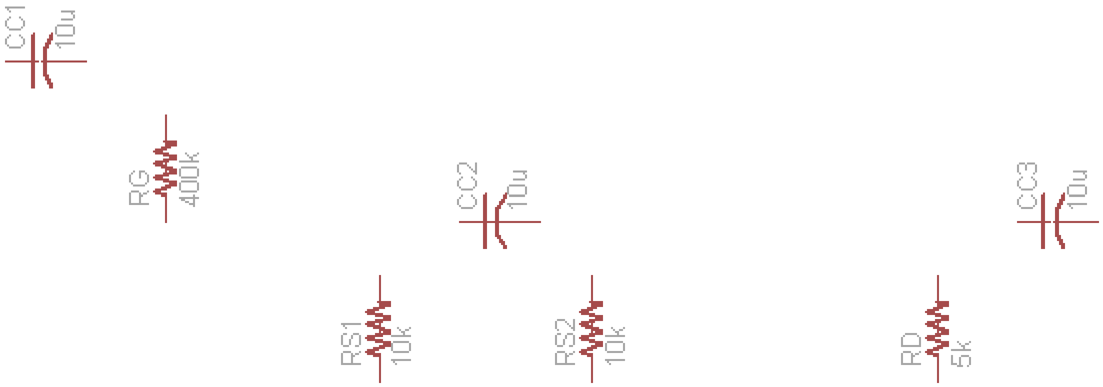
CAPACITOR, American symbol

Package: C050-035X075

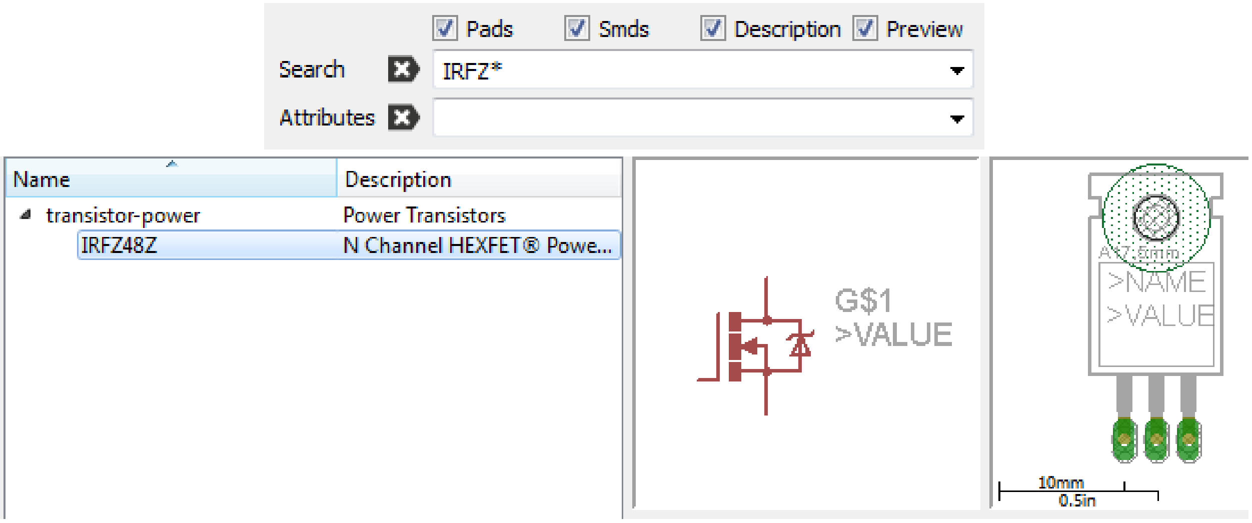
CAPACITOR

grid 5 mm, outline 3.5 x 7.5 mm

Selecting this symbol for all capacitors and adding the values:



Next, we will select the MOSFET. Normally, this would be done by searching datasheets to find a MOSFET with parameters matching those small signal parameters used in the design portion of the problem. The procedure would be similar to that used to select diodes, capacitors, and so on in our earlier design assignments. For the tutorial, we are going to assume that we will use a power-type MOSFET, similar to those from Lab 5. For the tutorial circuit, we will select the IRFZ44N as an example, by searching IRFZ* (searches all devices with the prefix 'IRFZ'):



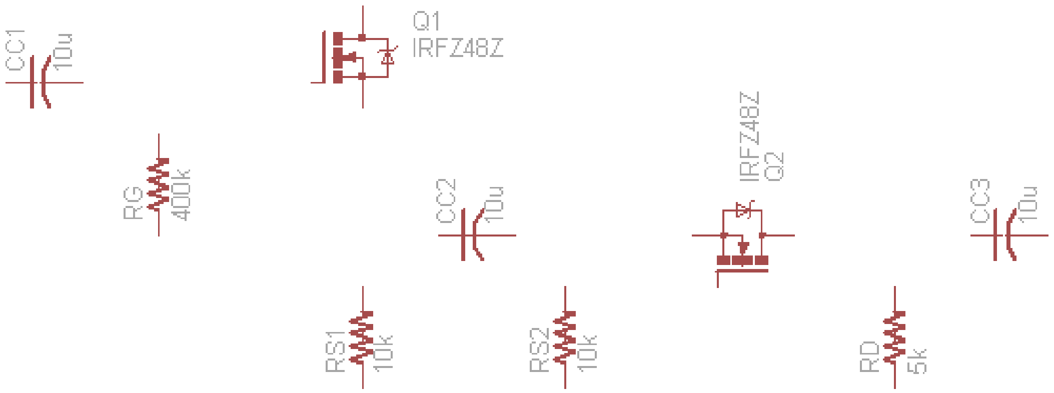
☑ Pads ☑ Smds ☑ Description ☑ Preview

Search: IRFZ*

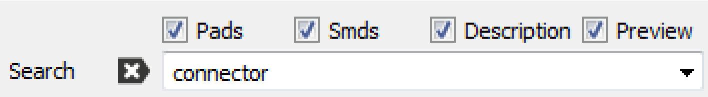
Attributes:

Name	Description
transistor-power	Power Transistors
IRFZ48Z	N Channel HEXFET® Powe...

Again, you may want to search for a more appropriate transistor type. A good indicator is the **case style.** The above uses the package type 'TO-220', which is typical of a power transistor. The 2N7000 and similar small-signal transistors used the physically smaller TO-92 case type. Note that for the above schematic symbol, the top pin is the drain, the bottom is source. Placing it for the two MOSFETs (note the rotation and mirroring for Q2):



While this might seem to be all of the components, we must also consider that the input, output, and power connections must be included. Since this is a self-contained circuit, we will include plugs or connectors on the board to allow easy connection to other circuits. By searching for 'connector':



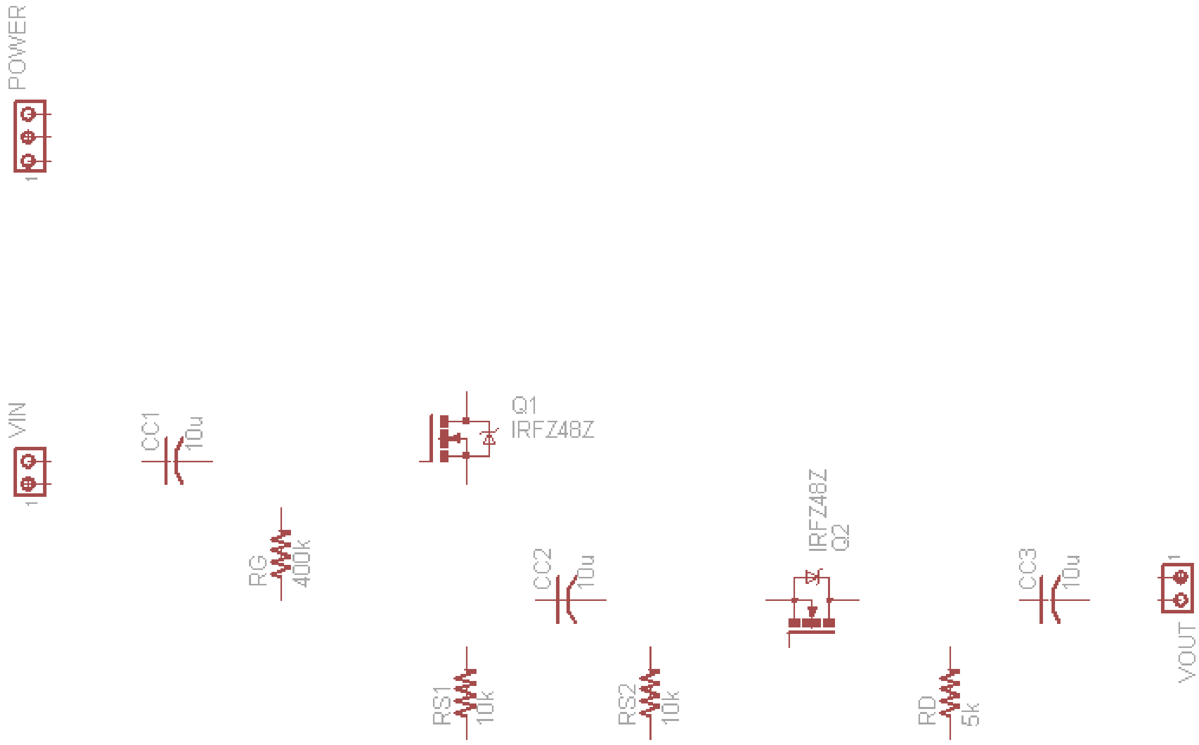
☑ Pads ☑ Smds ☑ Description ☑ Preview

Search ☒ connector

You will be presented with many categories of connectors to choose from – they come from a variety of manufacturers. Again, in a real-world design you would choose a connector beforehand and standardize it throughout the project. For this design assignment, we will leave the choice of connector up to you. For instance, you could use one 5-pin connector for this project $(+10\,V,\ -10\,V,\ Ground,\ v_{in},\ v_{out})$, or some combination of 2-pin and 3-pin connectors. We have chosen to create three for convenience of later connection: 'Power' (3-pin), 'Input' (2-pin), and 'Output' (2-pin). Given this choice, we choose appropriate connectors; we suggest checking the following categories:

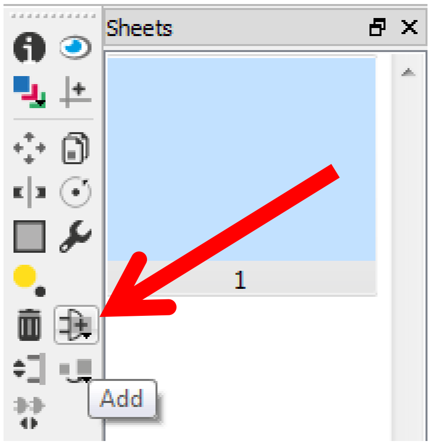
- con-amp → MTA0X (MTA02, MTA03, etc.) : Standard X-pin vertical connectors
- con-amp-mt → L0XP (L02P, etc.) : Standard X-pin horizontal/right-angle connectors
- con-amp-quick → M0X (M02, M03, etc.): Quick-connect connectors
- binding-post → The banana plug posts from the reference designs

You can choose any suitable through-hole connector that you wish. For the example, we selected MTA-type connectors:

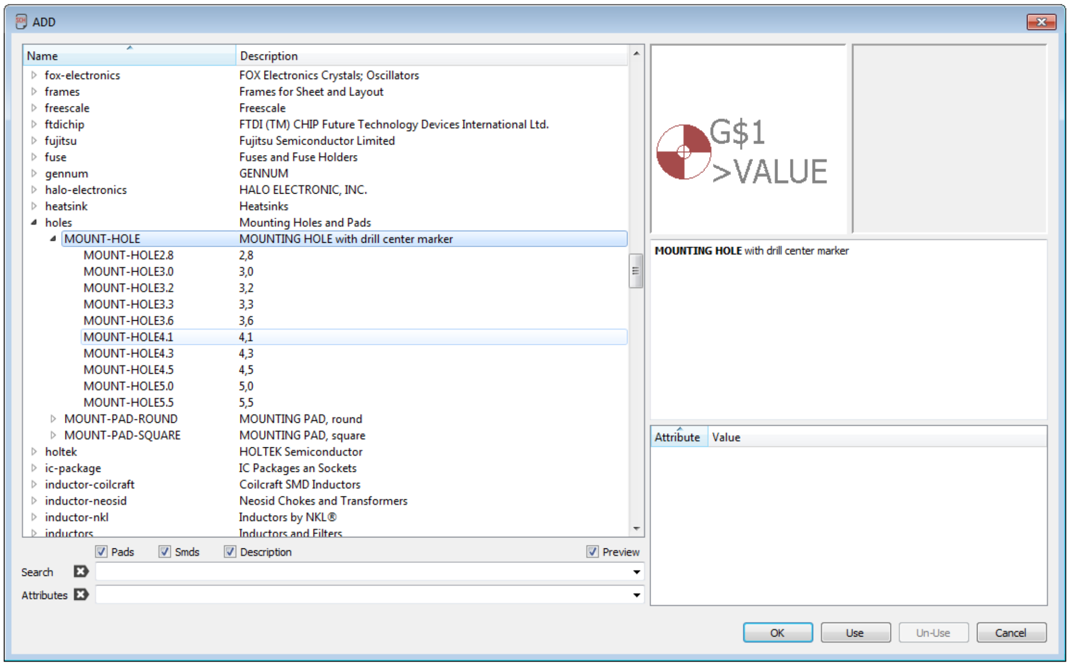


Adding Holes

In event that you want to have mounting holes in your board, again go to the 'Add Parts' icon on the tool panel:



Search for the size holes that you require and place them onto your diagram.



Name	Description
fox-electronics	FOX Electronics Crystals; Oscillators
frames	Frames for Sheet and Layout
freescale	Freescale
ftdichip	FTDI (TM) CHIP Future Technology Devices International Ltd.
fujitsu	Fujitsu Semiconductor Limited
fuse	Fuses and Fuse Holders
gennum	GENNUM
halo-electronics	HALO ELECTRONIC, INC.
heatsink	Heatsinks
holes	Mounting Holes and Pads
MOUNT-HOLE	MOUNTING HOLE with drill center marker
MOUNT-HOLE2.8	2,8
MOUNT-HOLE3.0	3,0
MOUNT-HOLE3.2	3,2
MOUNT-HOLE3.3	3,3
MOUNT-HOLE3.6	3,6
MOUNT-HOLE4.1	4,1
MOUNT-HOLE4.3	4,3
MOUNT-HOLE4.5	4,5
MOUNT-HOLE5.0	5,0
MOUNT-HOLE5.5	5,5
MOUNT-PAD-ROUND	MOUNTING PAD, round
MOUNT-PAD-SQUARE	MOUNTING PAD, square
holtek	HOLTEK Semiconductor
ic-package	IC Packages an Sockets
inductor-coilcraft	Coilcraft SMD Inductors
inductor-neosid	Neosid Chokes and Transformers
inductor-nkl	Inductors by NKL®
inductors	Inductors and Filters

☑ Pads ☑ Smds ☑ Description ☑ Preview

Search

Attributes

MOUNTING HOLE with drill center marker

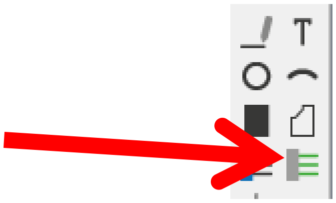
Attribute	Value

OK Use Un-Use Cancel

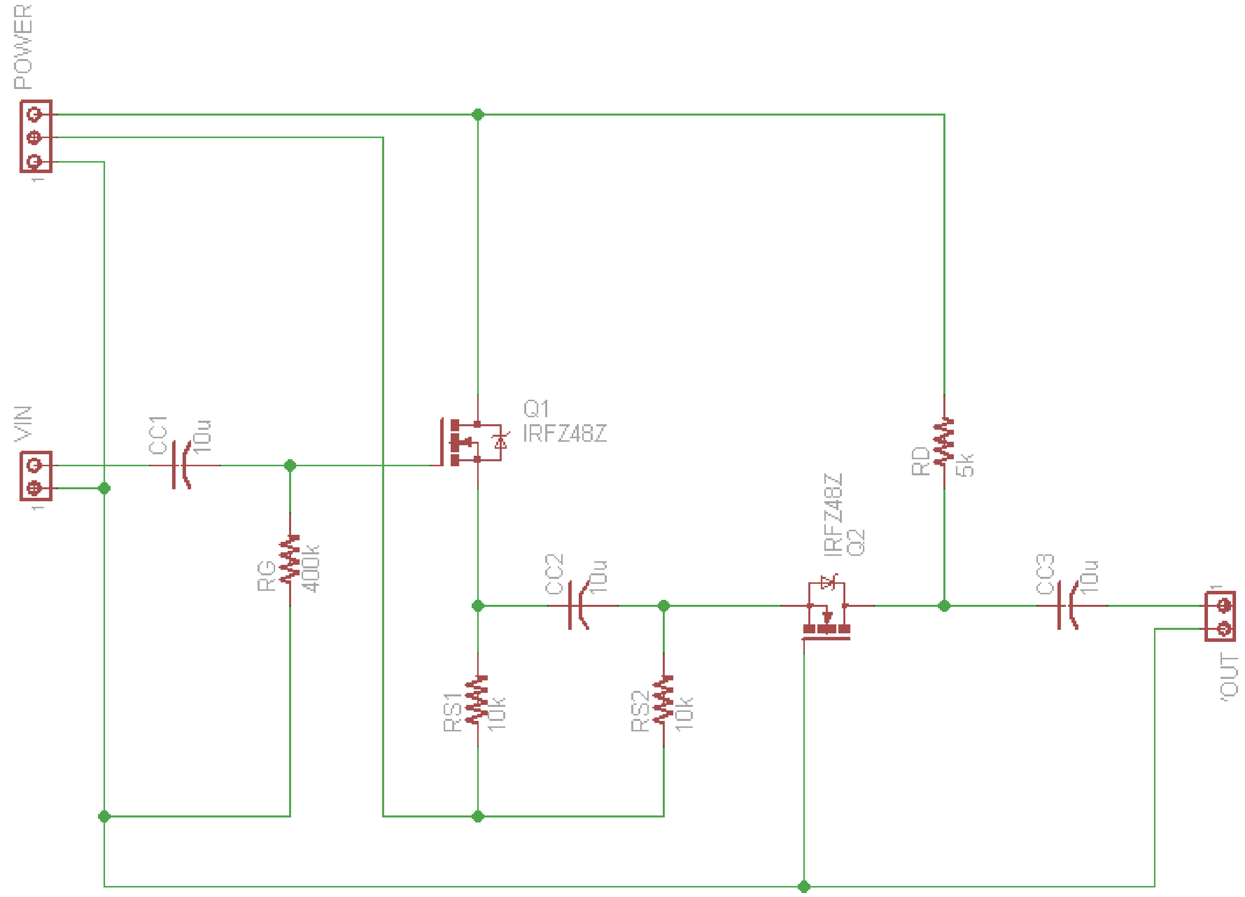
These holes will have to be positioned when you create your PCB.

Connecting the components

The last step is to use the 'net' tool to connect all parts together:



The result is as follows:

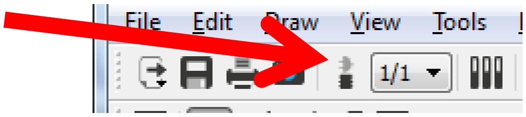


One should notice that:

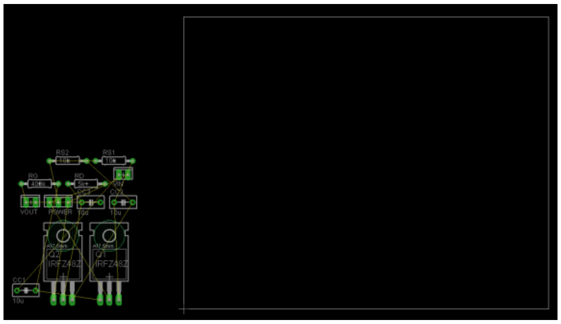
- The VIN and VOUT connectors each have a connection to ground, to make it convenient for the user to connect a load and source to the circuit.
- The bottom pin of the POWER connector is also ground, the middle pin is $(-10\,V)$, and the top pin is $(+10\,V)$ – note that we flipped resistor R_D for convenience.

PCB Creation

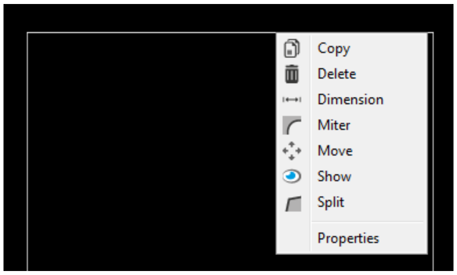
This schematic is ready for PCB creation. Save your schematic if you have not done so already, and **include a copy or printout in your deliverable**. Next, click the PCB creation button on the top toolbar:

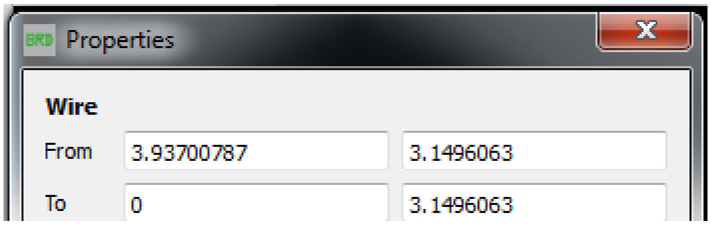


Answer 'Yes' to the dialog (if any), and then the PCB layout window should load. The result will be an unsorted set of parts (left) and a blank PCB area (right, bounded by white box).



We will need to move the parts into position. You may notice the yellow lines joining various pins of the parts; these represent connections ('nets') from the schematic that we just created. They will need to be physically realized by copper 'traces' that we add to the schematic. First, let's reduce the size of the board area, as we do not want to waste space on the finished board. Right-click on the white lines of the box and select 'Properties':

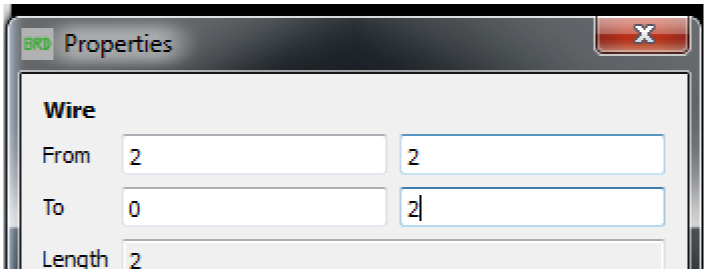



Properties

Wire

From	3.93700787	3.1496063
To	0	3.1496063

Change the coordinates of each 'wire' in the properties window to form a **2 *in*** × **3 *in*** square; this is the available PCB area for your board. You will need to change the properties of more than one line to do so. Note that the pictures which follow actually show a 2 × 2 *in* board.



Wire

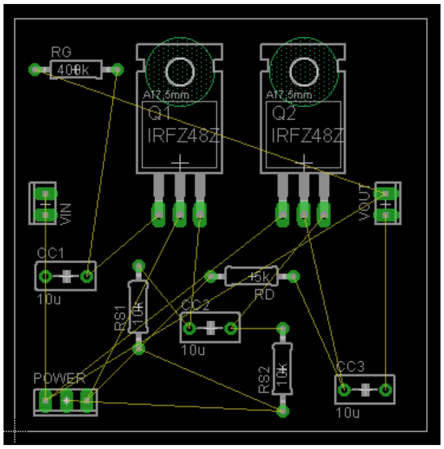
From 2 2

To 0 2

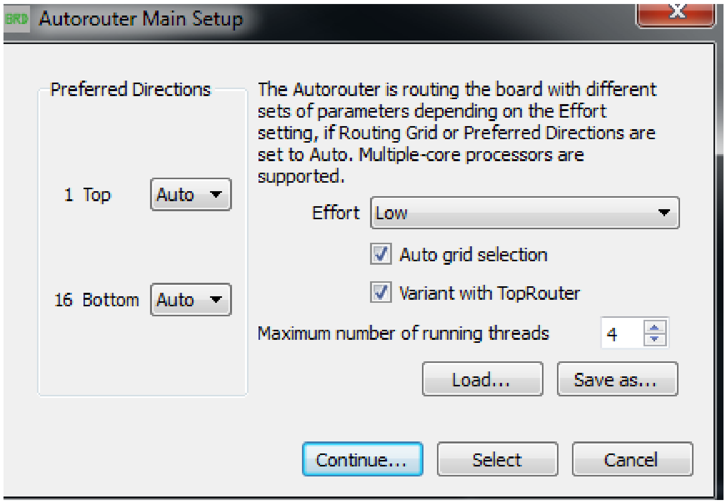
Length 2

These lines define the outline of your board and will eventually be used by the PCB manufacturer as well as the Eagle Board editor. To add a radius to the edges of your board, click the 'Miter' icon and enter a miter radius in the input menu: Radius: 0.2. Click on the corners of the board outline to create the radius.

Next, use the 'move', 'rotate', and 'mirror' tools (same toolbar locations as before) to move the parts into roughly the same layout as the schematic. You may want to make some optimizations to this placement – try to minimize the number of places that the yellow lines cross, for example.



Once ready, we could manually add copper traces to make each of the indicated connections, one-by-one. For complex, high-performance, or specialized circuits this may often be necessary. For simple circuits like this one, though, we can use the auto-router. From the tools menu, select 'Tools' → 'Autorouter…'



Autorouter Main Setup

Preferred Directions

1 Top: Auto

16 Bottom: Auto

The Autorouter is routing the board with different sets of parameters depending on the Effort setting, if Routing Grid or Preferred Directions are set to Auto. Multiple-core processors are supported.

Effort: Low

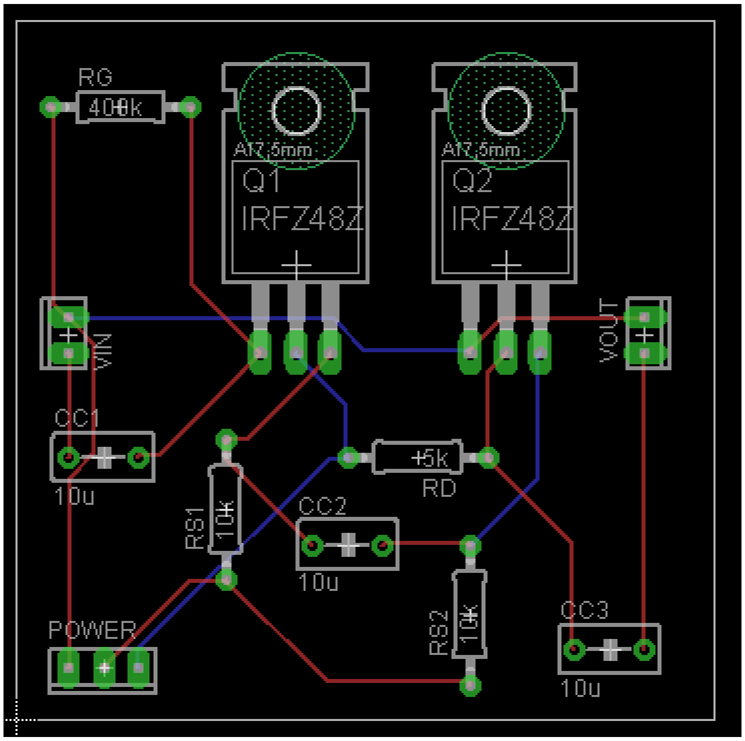
☑ Auto grid selection

☑ Variant with TopRouter

Maximum number of running threads: 4

This process will attempt to automatically route the necessary circuit traces. You can select 'High' for effort if you want a better routing result, but it may take longer. When ready, click 'Continue…' and then 'Start' on the dialog that is shown next. The process will then start. If you receive errors, you may need to go back and change your initial parts placement by clicking 'Cancel'. Otherwise, once the process ends, click 'End Job' to accept the layout.

The result should be similar to that shown on the next page.



In the previous picture, one can notice:

- The white outlines and text are called the **silkscreen** – these labels will be **printed in ink** onto the finished board to assist with assembly and troubleshooting.
- The green outlines represent **pads**, locations where component leads will be soldered.
- The red and blue lines represent **copper traces**, the electrical connections of your circuit. This is a **two-layer** board, meaning that it has copper traces on the top and bottom 'layers'. Multi-layer boards (with more than two layers) are possible; they are created by sandwiching traces or planes of conducting material between layers of insulating material. They are usually significantly more expensive, and are reserved for specialized equipment.
- Note that wherever a pad (green) is located, there is a connection from the top to bottom layer; you may also (depending on your parts placement) notice **vias**, which are separate connections from the top to bottom layer. In total, the blue and red lines, plus connections, should provide the same electrical connectivity as your original circuit.

In real-world design tasks, we would carefully optimize the placement of parts to minimize size, reduce noise (due to closely-located traces), and so on.

Generating Gerber Data with the CAM Processor

Before your board can be manufactured, Gerber Data must be generated. This data is used by the manufacturer to create your board. In Eagle this process is defined as a CAM Processor job.

Eagle comes with a predefined CAM Processor job names *gerb274x.cam* that automates the output of the most common Extended Gerber data for double sided boards.

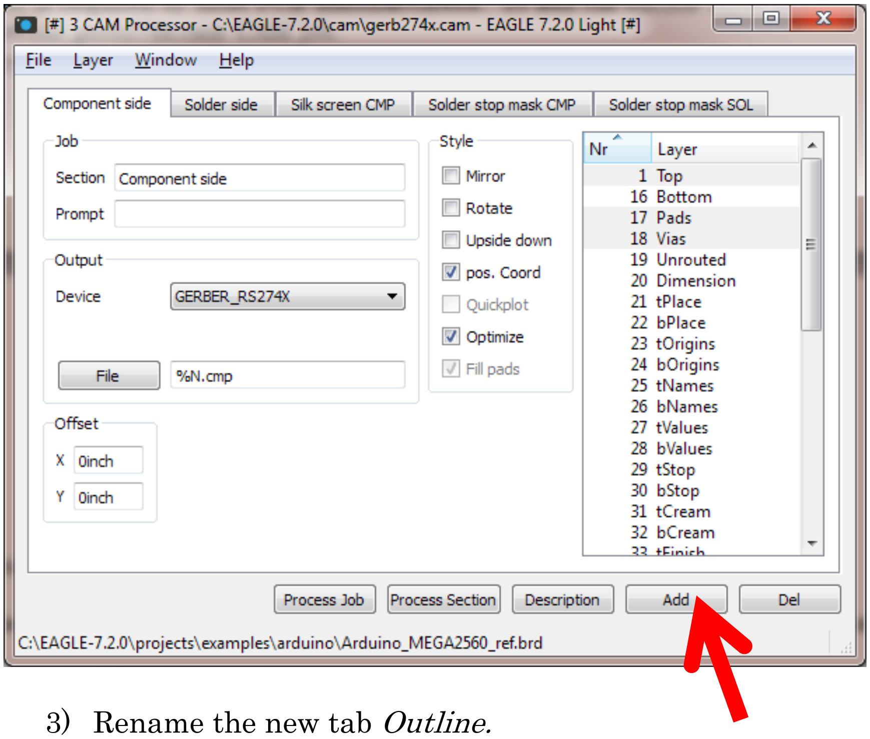
Please contact your board house to confirm which data is needed.

There are multiple ways to start the CAM Processor in Eagle.

1) You can load the job into the CAM Processor by double-clicking the entry with the name *gerb274x.cam* in the Control Panel's tree view (*CAM Jobs*). Open your board design by selecting *File>Open>Board* in the CAM Processor.
2) Clicking the CAM Processor icon 📷 in either the Board or Schematic editor window and selecting *gerb274x.cam* in the file dialog (*File>Open>Job*).
3) By selecting File>CAM Processor from the menu in either the Board or Schematic editor.

The default *gerb274x.cam* job does not include an outline layer output by default. This outline layer is required by most PCB manufacturers. To add the outline layer output:

1) Start the *gerb274x.cam* CAM job.
2) Copy the *Component Side* tab by clicking *Add.*



[#] 3 CAM Processor - C:\EAGLE-7.2.0\cam\gerb274x.cam - EAGLE 7.2.0 Light [#]

Job
Section: Component side
Prompt:

Output
Device: GERBER_RS274X
File: %N.cmp

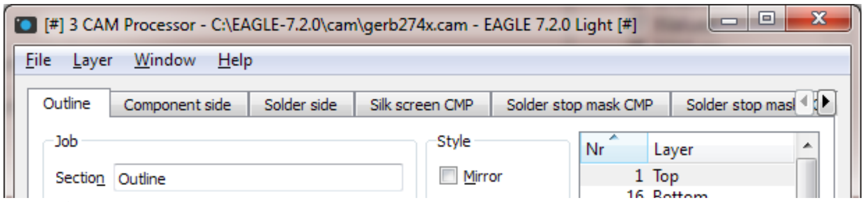
Offset
X: 0inch
Y: 0inch

Style
☐ Mirror
☐ Rotate
☐ Upside down
☑ pos. Coord
☐ Quickplot
☑ Optimize
☑ Fill pads

Nr	Layer
1	Top
16	Bottom
17	Pads
18	Vias
19	Unrouted
20	Dimension
21	tPlace
22	bPlace
23	tOrigins
24	bOrigins
25	tNames
26	bNames
27	tValues
28	bValues
29	tStop
30	bStop
31	tCream
32	bCream
33	tFinish

C:\EAGLE-7.2.0\projects\examples\arduino\Arduino_MEGA2560_ref.brd

3) Rename the new tab *Outline.*



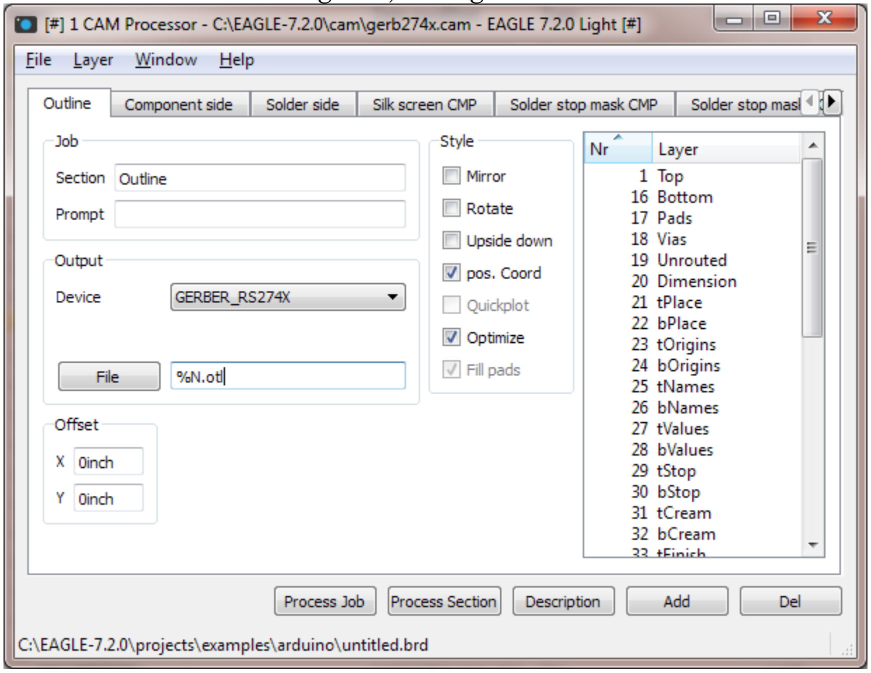
[#] 3 CAM Processor - C:\EAGLE-7.2.0\cam\gerb274x.cam - EAGLE 7.2.0 Light [#]

Job
Section: Outline

Style
☐ Mirror

Nr	Layer
1	Top
16	Bottom

4) De-select the *Top, Pads,* and *Vias* layers.
5) Select the *Dimension* layer. (This layer contains the board outline)
6) In the *File* dialog box, change the extension to *.otl.*



[#] 1 CAM Processor - C:\EAGLE-7.2.0\cam\gerb274x.cam - EAGLE 7.2.0 Light [#]

Job
Section: Outline
Prompt:

Output
Device: GERBER_RS274X
File: %N.otl

Offset
X: 0inch
Y: 0inch

Style
☐ Mirror
☐ Rotate
☐ Upside down
☑ pos. Coord
☐ Quickplot
☑ Optimize
☑ Fill pads

Nr	Layer
1	Top
16	Bottom
17	Pads
18	Vias
19	Unrouted
20	Dimension
21	tPlace
22	bPlace
23	tOrigins
24	bOrigins
25	tNames
26	bNames
27	tValues
28	bValues
29	tStop
30	bStop
31	tCream
32	bCream
33	tFinish

C:\EAGLE-7.2.0\projects\examples\arduino\untitled.brd

To run the CAM Processor job, click the button *Process Job.* All the output files will be written into the directory where the board file is located.

The files have the following meanings:

demo3.cmp	Component side
demo3.sol	Solder side
demo3.plc	Silkscreen for component side
demo3.stc	Soldering mask for the component side
demo3.sts	Soldering mask for the solder side
demo3.gpi	Information file, not relevant here and can be deleted
demo3.otl	Outline information

These files need to be sent to your board manufacturer.

Generating Drill Data

In addition to the previoiusly generated Gerber files, PCB manufacturers require an EXCELLON drill file. This file contains data about the location and size of holes to be drilled in the PCB. Drilling data can be generated by opening the job *excellon.cam* in the CAM Processor. This job consists of one single step. The EXCELLON device generates a file that contains both drill data and drill table. The output file has the file extension *.drd*. This file must also to be sent to your board manufacturer.

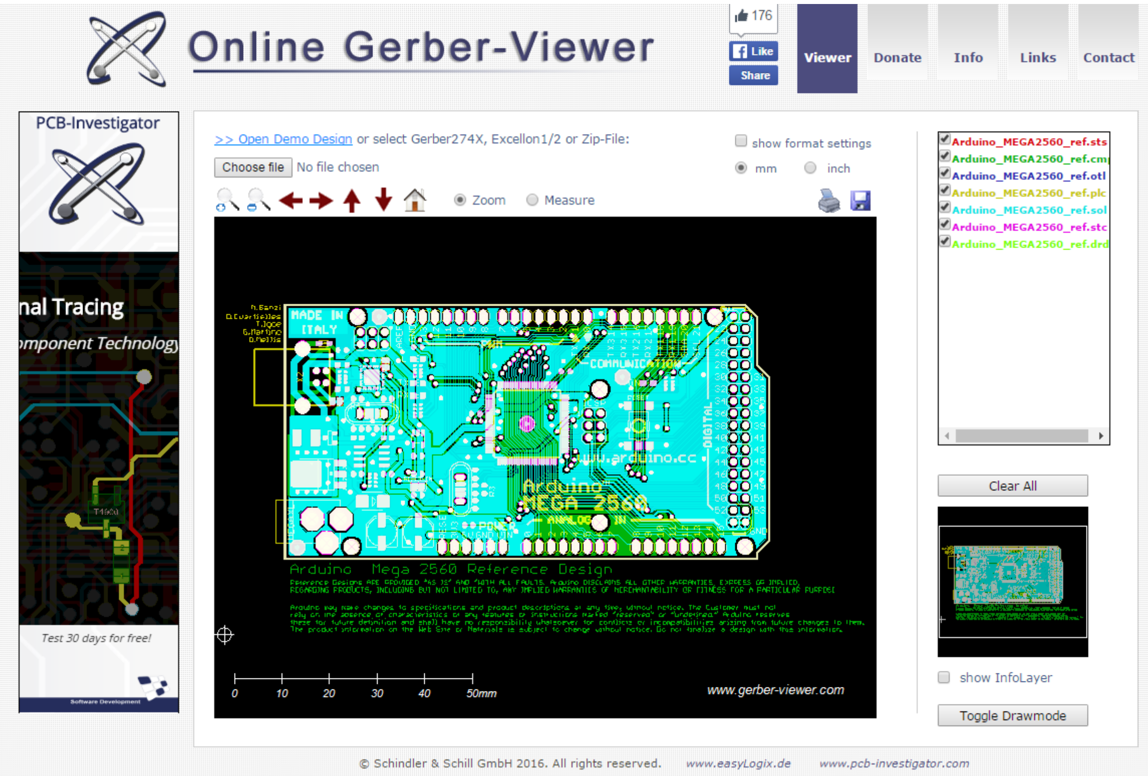
A final submission to a PCB manufacturer (or for your assignment) should be a ZIP file containing all necessary Gerber and EXCELLON files.

Further information can be found on the CAM Processor help pages and in the EAGLE manual.

(Reference: EAGLE Version 7 – Tutorial)

Viewing Gerber Data

To ensure that all your manufacturing files were created properlay and everything lines up properly, you can view them using a Gerber Viewer. You use the free online Gerber viewer at http://www.gerber-viewer.com/. Simply upload your complete ZIP file to the site and select the layer you wish to view. Ensure that everything is as expected and then submit your design.



>> Open Demo Design or select Gerber274X, Excellon1/2 or Zip-File:

Choose file No file chosen

☐ show format settings

● mm ○ inch

● Zoom ○ Measure

☑ Arduino_MEGA2560_ref.sts
☑ Arduino_MEGA2560_ref.cm
☑ Arduino_MEGA2560_ref.otl
☑ Arduino_MEGA2560_ref.plc
☑ Arduino_MEGA2560_ref.sol
☑ Arduino_MEGA2560_ref.stc
☑ Arduino_MEGA2560_ref.drd

☐ show InfoLayer

© Schindler & Schill GmbH 2016. All rights reserved. www.easyLogix.de www.pcb-investigator.com